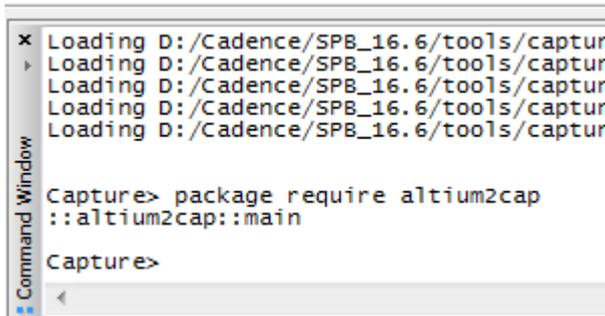


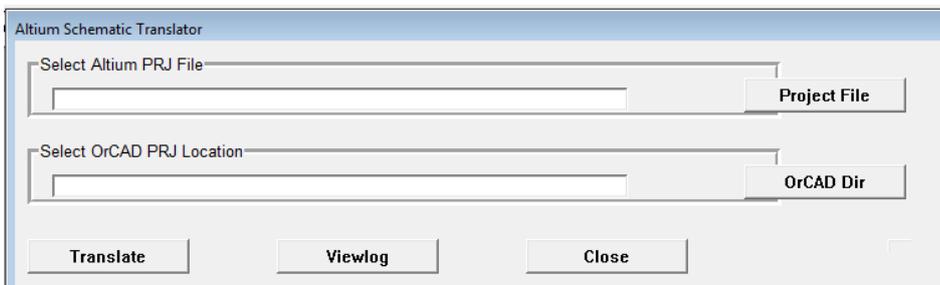
## Running the Altium to OrCAD Capture translator

Turn on the OrCAD TCL command window under the view command window menu. In the command window enter these commands:

```
package require altium2cap  
::altium2cap::main
```

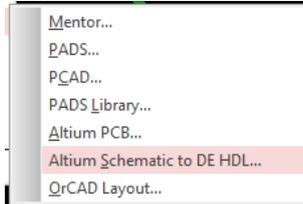


Browse to the ACSII PrjPCB Altium design file to be translated. See note below on exporting the ASCII version of the Altium design. Then select the output project directory and click Translate.



## Running the Altium to DE HDL schematic translator

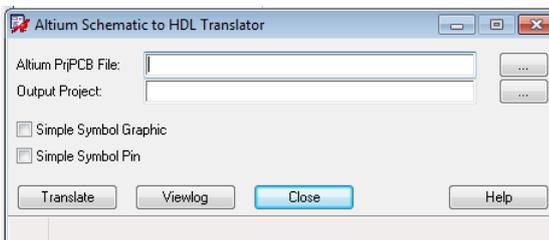
In Allegro PCB Editor under the file menu choose import, Cad Translators, Altium Schematic to DE HDL.



Browse to the ACSII PrjPCB Altium design file to be translated. See note below on exporting the ASCII version of the Altium design. Then select the output project directory.

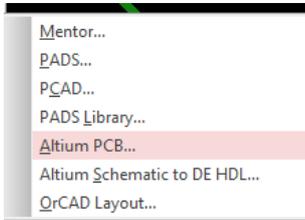
By default the translator tries to match the look of the original schematic. There are 2 options to control the graphics:

1. Simple Symbol Graphic. Checking this will ignore any custom symbol colors defined in Altium.
2. Simple Symbol Pin. Checking this will ignore any custom pin shapes defined in Altium.



## Running the Altium to Allegro PCB Editor translator

In Allegro PCB Editor under the file menu choose import, Cad Translators, Altium PCB.



Browse to the ASCII PcbDoc Altium design file to be translated. See note below on exporting the ASCII version of the Altium design.



When the "Create Individual Symbol Definitions" box is checked, the translator will generate a separate symbol definition for each instance of a component by adding a suffix. For example 0805\_1, 0805\_2, 0805\_3 and so on. This allows to account for instance specific footprint modifications within the Altium design.

By default this option is unchecked, which means that the translator will create one symbol definition only for a given Altium footprint. Use this option only if instance specific changes have been made in the Altium design.

## Translating a schematic and PCB design project

The process to translate an entire Altium project (schematic and PCB) is to first translate the schematic into either DE HDL or OrCAD. Then from the translated schematic export an Allegro netlist into a new empty Allegro database. Then run the Altium to Allegro PCB translator from that Allegro database. This way the Altium2PCB translator will consider the netlist and device logic already imported and start to translate only the remaining data in order to complete the board.

## Exporting the Altium Project to ASCII

To translate Altium Project the Altium schematic and PCB database (.SchDoc & .PcbDoc) needs to be in ASCII Format which is accessible under the file save data type in Altium. Based on experience Altium Version 6 ASCII files work best for OrCAD Translation.

